International Conference on Mechanical Engineering Research (ICMER2013), 1-3 July 2013
Bukit Gambang Resort City, Kuantan, Pahang, Malaysia
Organized by Faculty of Mechanical Engineering, Universiti Malaysia Pahang
Paper ID: P135

COMPUTATIONAL ANALYSIS OF AN EFFECT OF AERODYNAMIC PRESSURE ON THE SIDE VIEW MIRROR GEOMETRY

P. Murukesavan[1], M. A. Nik Mu'tasim[1*] and I. Mat Sahat[1]

[1]Faculty of Mechanical Engineering, University Malaysia Pahang
26600 Pekan, Pahang, Malaysia
*Email: ammar@ump.edu.my
Phone : +6094246264; Fax : +6094242202

ABSTRACT

This paper describes the evaluation of aerodynamic flow effects on side mirrors geometry for a passenger car using ANSYS Fluent CFD simulation software. Result from analysis of pressure coefficient on the side view mirror designs is evaluated to analyze the unsteady forces that cause fluctuations to mirror surface and image blurring. The fluctuation also causes drag forces that increase the overall drag coefficient, with an assumption resulting in higher fuel consumption and emission. Three Features of side view mirror design were investigated with two input velocity parameters of 17 m/s and 33 m/s. Results indicate that the half-sphere design shows the most effective design with less pressure coefficient fluctuation and drag coefficient.

***Keywords*:** Computational fluid dynamics (CFD); side view mirror; pressure coefficient.

INTRODUCTION

Reducing fuel consumption, and therefore reducing the carbon dioxide emissions, is one of the most important goals in today's car industry. One way this can be achieved is by reducing the engine size, using an electric motor with a combustion engine, reducing the weight of the car and reducing the aerodynamic drag of the car. The latter is of great importance when it comes to velocity over 60km/h. Above this velocity, the aerodynamic resistance is higher than the rolling resistance (Versteeg, 2007). Streamlined body design in a passenger car helps reducing the aerodynamic drag and eventually improves the engine mileage. On the contrary, accessories attached to the body skin of a car cause the unfavourable aerodynamic examples. In order to obtain the rear sight, unfortunately the mirror does not pay only the aerodynamic penalty which increases drag from the body, but also causes the acoustic noise thus causes mirror fluctuations to the cabin crews. While the aerodynamic body styling of the passenger car has been upgraded with a lot of efforts, the defects caused by important accessory such as the side view mirror have been ignored. The main stream meets a side flow which has the flow direction tangent to the windshield surface near the A-pillar. And a conical vortex sheet is generated along the pillar and merges into the mainstream. Therefore, very complicate flow pattern appears by combining these flow patterns near the driver side window. Moreover, since the side mirror is mounted on the driver door near hinge, the wake flow behind this obstacle become much complicated. (Gillespie, 2000).

Numbers of research has been done to study the effect of flow and noise towards the front side view mirrors. Some studies have been done experimentally using pressure measurement devices on automobile side mirror to evaluate the effect of pressure fluctuations on mirror vibrations (Jaitlee et al., 2004). Same approach was also done by using hot wire anemometry and the pressure scanning system in a wind tunnel at high Reynolds Number (Kim et al., 2008). This resulted in a vortex sheet with a conical shape developed between the centres of the side mirrors. Studies conducted using computational fluid dynamics on the noise generation on side mirrors have been done using Detached Eddy Simulation, DES, and different turbulent methods (Grahs et al., 2006). Another investigation was also done using the same method with the different approach by increasing the diameter of the side mirror in a high constant Reynolds Number (Ask 2006). Three turbulent models were studied on flow around two mirrors to measure the surface pressure fluctuations and resulted in underestimated sound pressure levels along the side mirrors. (Snegirev et al., 2007).

MATHEMATICAL MODEL

The K-epsilon model is one of the most common turbulence models, although it just doesn't perform well in cases of large adverse pressure gradients. It is a two equation model that means, it includes two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy. K-epsilon model has been shown to be useful for free-shear layer flows with relatively small pressure gradients. Similarly, for wall-bounded and internal flows, the model gives good results only in cases where mean pressure gradients are small; accuracy has been shown experimentally to be reduced for flows containing large adverse pressure gradients.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial t_j}(\rho k u_j)=\frac{\partial}{\partial x_j}\left[(\mu+\frac{\mu_t}{\mu_k})\frac{\partial k}{\partial x_j}\right]+P_k+P_b-\rho\in-Y_M+S_k \tag{1}$$

$$\frac{\partial}{\partial t}(\rho\in)+\frac{\partial}{\partial x_j}(\rho\in u_j)=\frac{\partial}{\partial x_j}\left[(\mu+\frac{\mu_t}{\mu_\in})\frac{\partial\in}{\partial x_j}\right]+\rho C_1 S_\in-\rho C_2 S_\in-\rho C_2\frac{\in^2}{k+\sqrt{v\in}} + C_{1\in}\frac{\in}{k}C_{3\in}P_b+S_\in \tag{2}$$

Where

$$C_1=\max[0.43,\frac{n}{n+5}],n=S\frac{k}{\in},S=\sqrt{S_iS_i} \tag{3}$$

In these equations, P_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, calculated in same manner as standard k-epsilon model. P_b is the generation of turbulence kinetic energy due to buoyancy, calculated in same way as standard k-epsilon model.

DESIGN AND MODELING

Wind Tunnel

The wind tunnel is considered is made up of a straight floor and roof with an inlet up stream of the car 8.145m distance ahead of the front of the car and the outlet 20m downstream of the car. So that, as the tunnel outlet is far from the car, it can be assumed that it will have a little effect on the air flow close to the car. The tunnel height is taken a 5m. Then the boundaries if the simulation model is the car surface, side mirror, velocity inlet, pressure outlet, symmetry, symmetry upper, symmetry side and road.

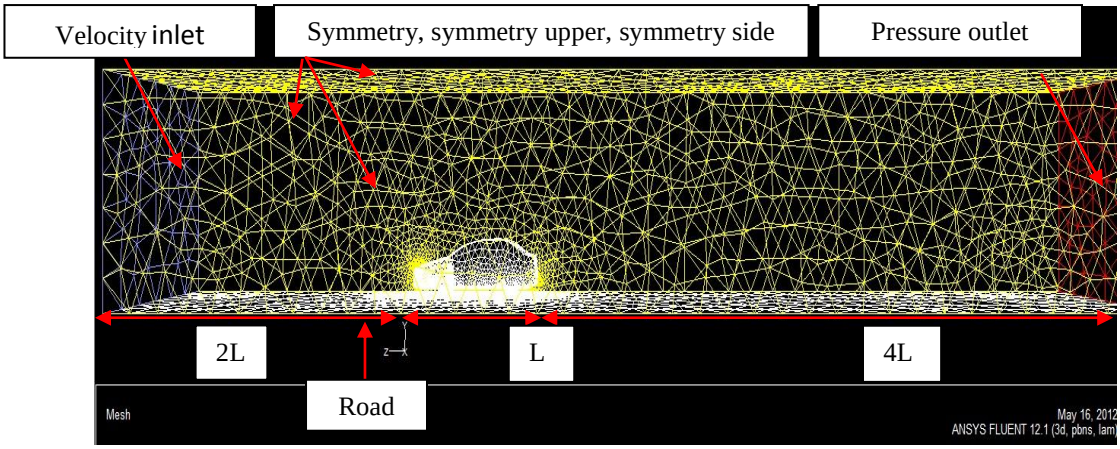


Figure 1. Wind Tunnel Simulation Setup

Generic Side Mirror Designs

The generic model of the side mirrors is shown in the Figure below with common dimensions. The model is using the actual common dimension of side mirror which is the length of the model is 1.310m, the width of the model is 0.540m, and the height of the model is 0.930m. Solid Works was used to produce the models with the required dimensions. Design 1 was done based on a semi-sphere shape while Design 2 was done based on a sharp end with triangular shape and the Design 3 was done by combining rectangular shape with triangular edges.

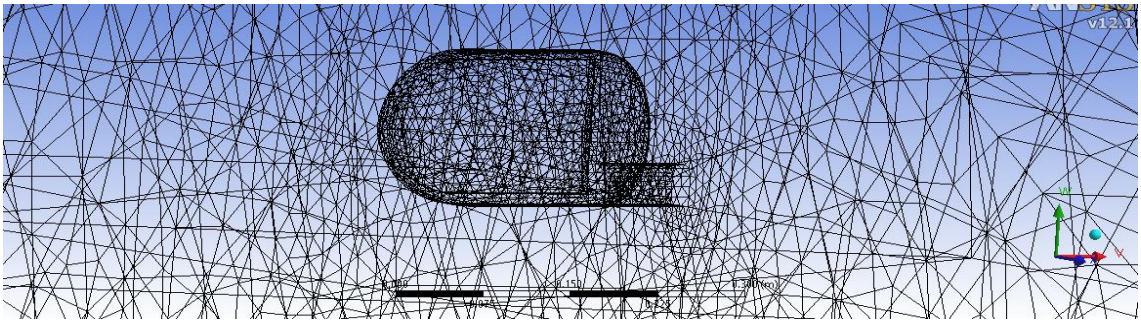
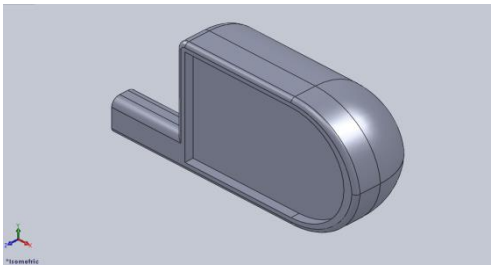
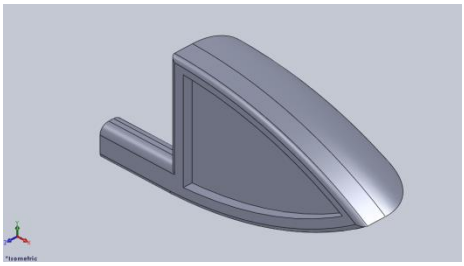


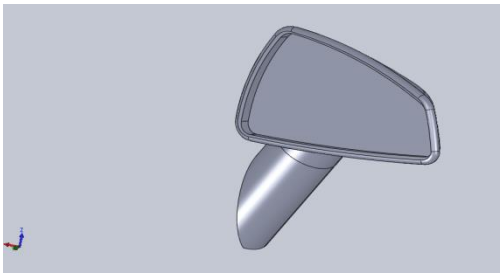
Figure 2. Meshing Generic Side Mirror Setup



(a)



(b)



(c)

Figure 3. Side Mirror Designs (a) Design 1 (b) Design 2 and (c) Design 3.

Figure below shows the selection of 100 critical points across the side mirror for all the designs used. This is done to extract the results which are pressure coefficient, total pressure, drag coefficient and lift coefficient at the required area. The critical points are chosen across the frontal area of the side mirror which faces the simulated flow.

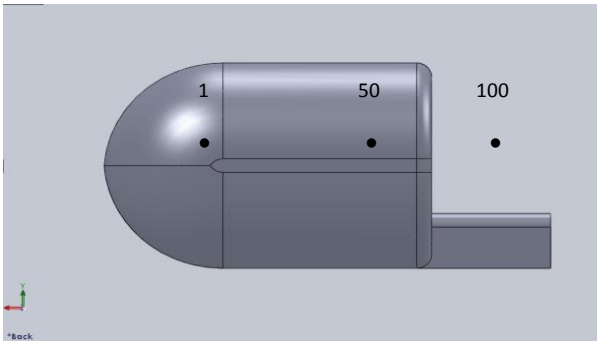


Figure 4. 100 critical points across the side mirror frontal area

VALIDATION

In order to validate with the same reference, an experiment analysis was conducted by (Kim 2008) with the same design 1 with a velocity of 25 m/s. Figure 5 shows comparison between the present works with the experimental results. It can be seen that a good agreement is achieved between the results obtain from computational and the experimental results for the mirror housing.

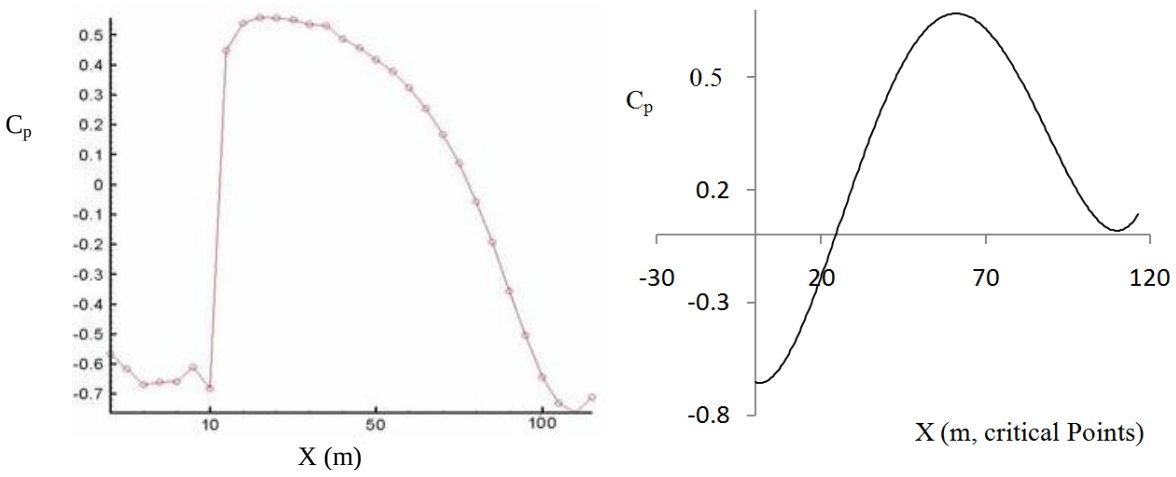
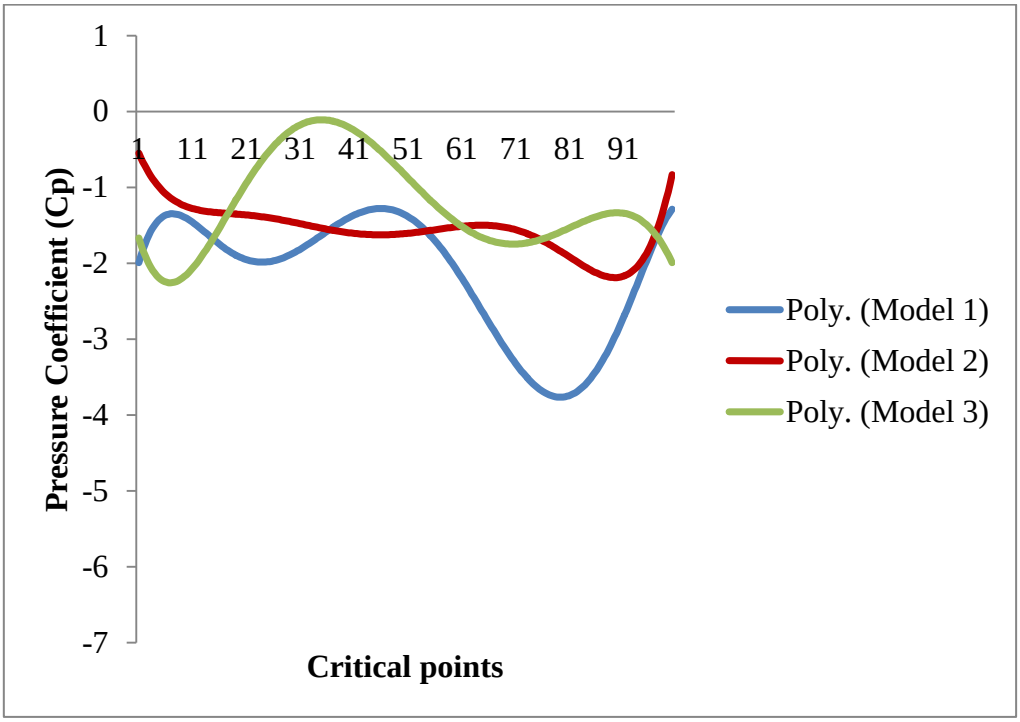


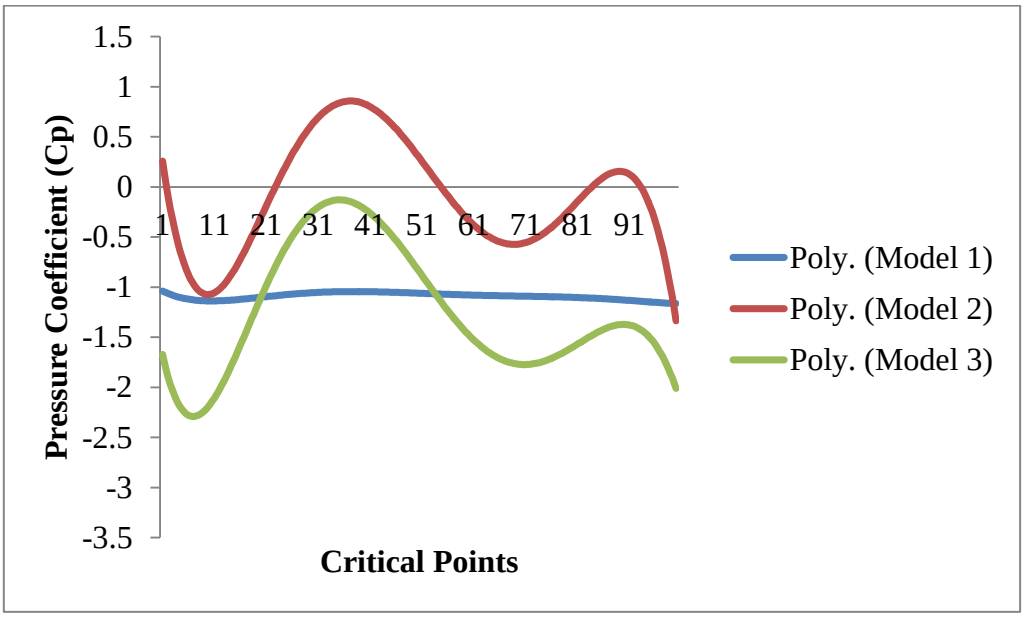
Figure 5. Comparison between the experimental results with Present Study for velocity of 25 m/s.

RESULTS AND DISCUSSION

The pressure coefficient at a point near a body is independent with the body size. To determine the model, it is tested in a wind tunnel or water tunnel, pressure coefficients is determined at critical locations around the model and these pressure coefficients is used with confidence to predict the fluid pressure at those critical locations around a passenger car.

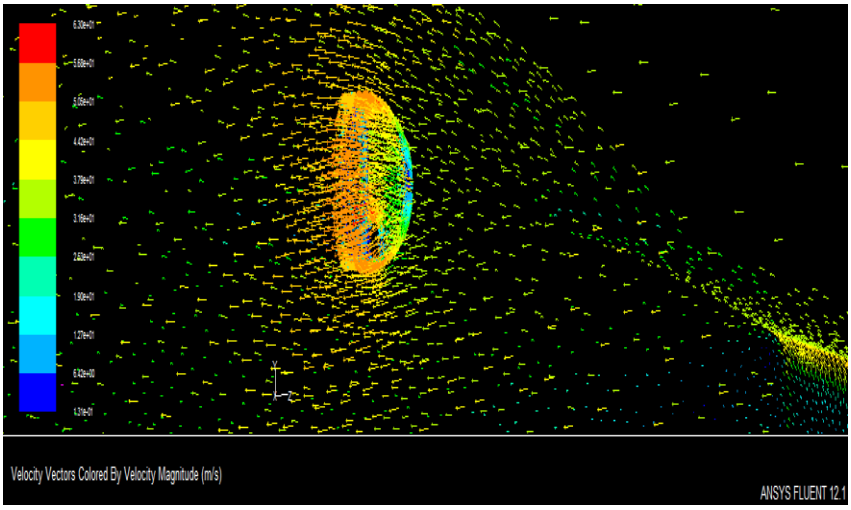


(a)

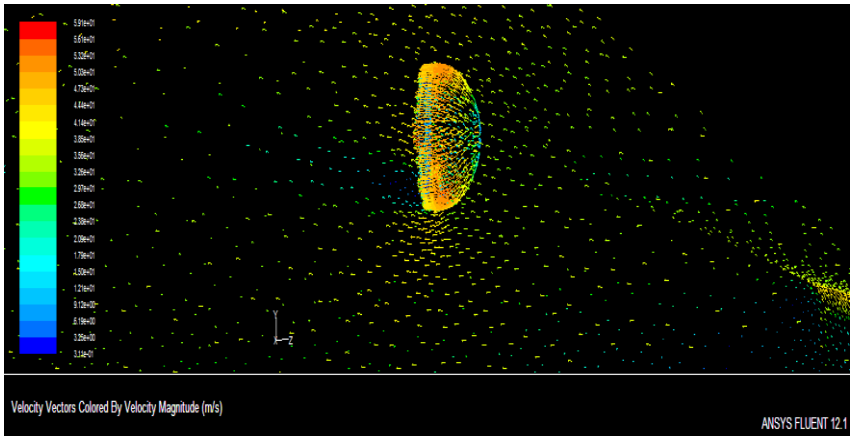


(b)

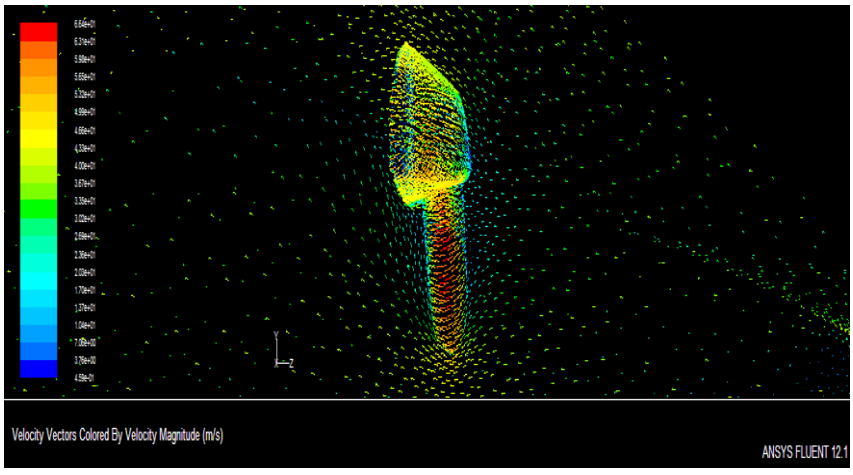
Figure 4: Side Mirror Designs velocity at (a) 16.7 m/s (b) 33.3 m/s



(a)



(b)



(c)

Figure 5: Side Mirror flow contour for (a) Design 1 (b) Design 2 and (c) Design 3

Before the simulation was done, boundary conditions such as environment pressure and outlet pressure of 1atm are applied. The speed used for the simulation is 16.67 m/s (60 km/h), 25 m/s (90 km/h) and 33.33 m/s (120 km/h). The speed used is same as the experiment method done by Alam et al., 2007. Non-slip condition was applied on the entire solid surface. The discussion done shows the pressure coefficient, total pressure distribution, drag coefficient and lift coefficient value around the car side mirror. Therefore the effect of different side mirrors designs based on the pressure coefficient, drag coefficient and lift coefficient can be estimated. For all the cases, the pressure coefficient around the passenger car was negative but that associated to in front of the side mirror which is always positive. The CFD simulation of flow over passenger car side mirror was in good agreement with that of the experiment method done by Alam et al., in 2007. The comparison of the drag coefficient respectively for the nine cases was studied. All the cases show positive results in a range value of 0.4 to 0.6. The pressure coefficients fluctuate based on the design and speed. Whereas, lift coefficient show the lift coefficient of the nine cases studied. All the cases show negative results in a range value of -0.05 to -0.3. Side Mirror 3 shows highest drag coefficient with the highest lift coefficient and Side Mirror 1 show the lowest drag coefficient with the smallest lift coefficient. Therefore, Mirror 1 is suitable to be used in passenger cars due to less mirror fluctuations which could lead to fuel saving.

CONCLUSION

This conclusion was made from the work presented here. Fluctuating aerodynamic pressures are not uniformly distributed over an automobile mirror surface. The highest magnitude of fluctuating pressure coefficient was found at the central across the back of the side mirror while the lowest magnitude of fluctuating pressure coefficient was found at the sharp edges around the side mirror frontal area. As the yaw angle is known to affect mirror noise and vibration, this should be considered in future work.

ACKNOWLEDGEMENTS

The authors would like to thank Universiti Malaysia Pahang (UMP) for financial support under RDU110379.

REFERENCES

Alam F., R. Jaitlee and S. Watkins, 2001, *Aerodynamic Effects on an Automotive Rear Side View Mirror*, School of Aerospace, Mechanical and Manufacturing Engineering RMIT University, Melbourne, VIC 3083, AUSTRALIA, 16th Australasian Fluid Mechanics Conference. 762-765

Heisler. H. 2002, *Advanced Vehicle Technology*. Elsevier Butterworth Heinemann, Jordan Hill, Oxford. Pg 121-123

Jaitlee R., F. Alam and S. Watkins, 2001, Pressure Measurements on an Automobile Side Rear View Mirror, School of Aerospace, Mechanical & Manufacturing Engineering RMIT University, Melbourne, VIC 3083, AUSTRALIA, 15th Australasian Fluid Mechanics Conference

Jeong-Hyun Kim, Yong Oun Han, Min Hwa Lee, In Ho Hwang, And Ui Hun Jung, Surface *Flow And Wake Structure Of A Rear View Mirror Of The Passenger*

Car, School Of Mechanical Engineering Yeungnam University, Gyeongsan, Gyeongbuk 712-749, Korea, Bbaa Vi International Colloquium On: Bluff Bodies Aerodynamics & Applications.

Jonas Ask and Lars Davidson, 2006, *The Sub-critical Flow past a Generic Side Mirror and its Impact on Sound Generation and Propagation*, 12th AIAA/CEAS Aero acoustics Conference, 8-10 May, Cambridge, Massachusetts

Martin Olsson, *Designing and Optimizing Side-View Mirrors*, Department of Applied Mechanics, Division of Vehicle Engineering and Autonomous *Systems*, Chalmers University Of Technology.

Michele Ferlauto, Roberto Marsilio, Computer & Fluids, 2006, *A Viscous Inverse Method for Aerodynamic Design*, pg 304-325

Snegirev A.Yu., S.V. Lupulyak, Yu.K. Shinder, *Use Of High Performance Computing For Simulations Of Aerodynamically Generated Noise In Turbulent Separated Flows*, Laboratory For Applied Mathematics And Mechanics Saint-Petersburg State Polytechnic University Polytechnicheskaya, West-East High Speed Flow Field Conference.

Thomas D. Gillespie. (2000). *Fundamentals of Vehicle Dynamics.* Pg 74-96.

Versteeg H. K. and W. Malalasekera. 2007, *An Introduction to Computational Fluid Dynamics, The Finite Volume Method*. Pearson Prentice Hall, second edition.

Yiping Wang, Zhengqi Gu, Weiping Li and Xiaohui Lin, 2010, *Evaluation of Aerodynamic Noise Generation by a Generic Side Mirror*, World Academy of Science, Engineering and Technology 61. 364-571.

Yunus A. Cengal, John M.Cimbala. 2006, *Fundamentals of Fluid Mechanics*, 1st Edition in SI Units. Pg 562-600